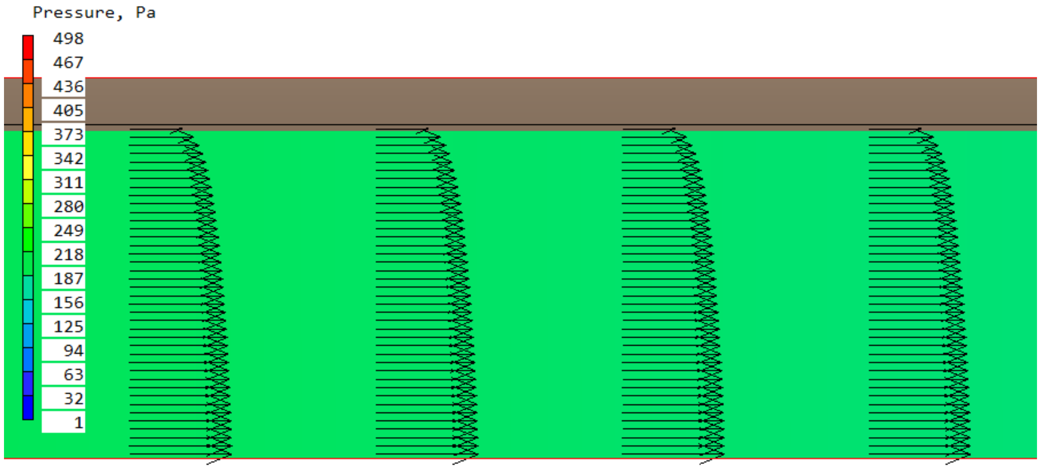


CFD test case 2
Turbulent water transport in a hydraulically smooth pipeline

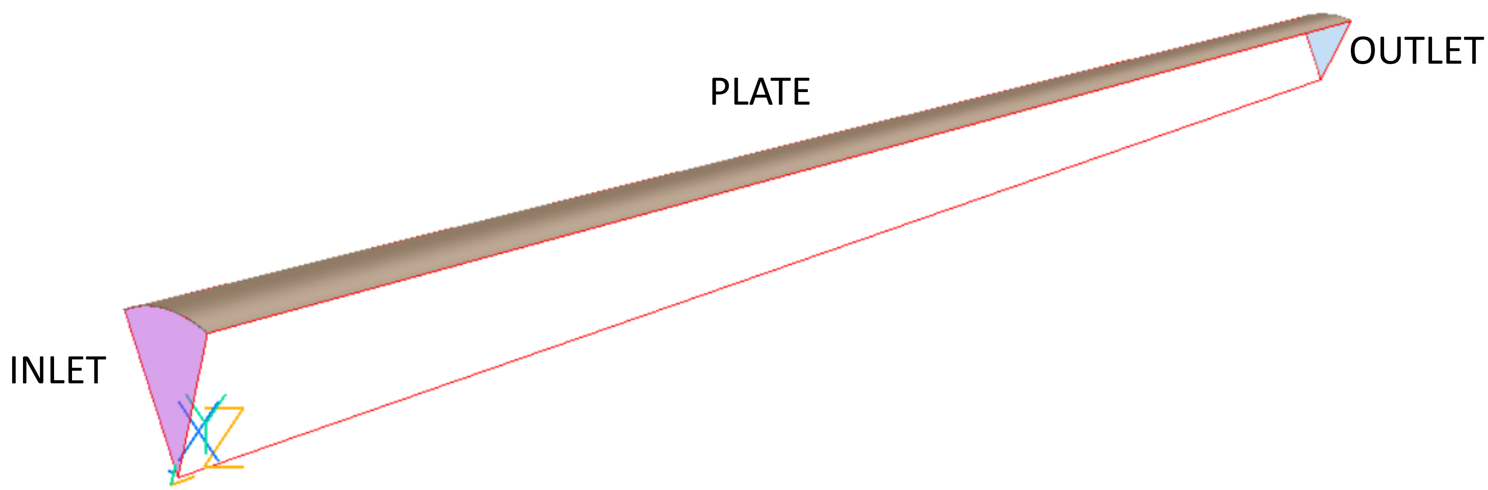
Suggestions for setting up the case

Ing. Luca Nicola Quaroni
Ing. Jorge Soto
Ing. Federico Lanteri
Prof. Gianandrea Vittorio Messa

POLITECNICO
MILANO 1863

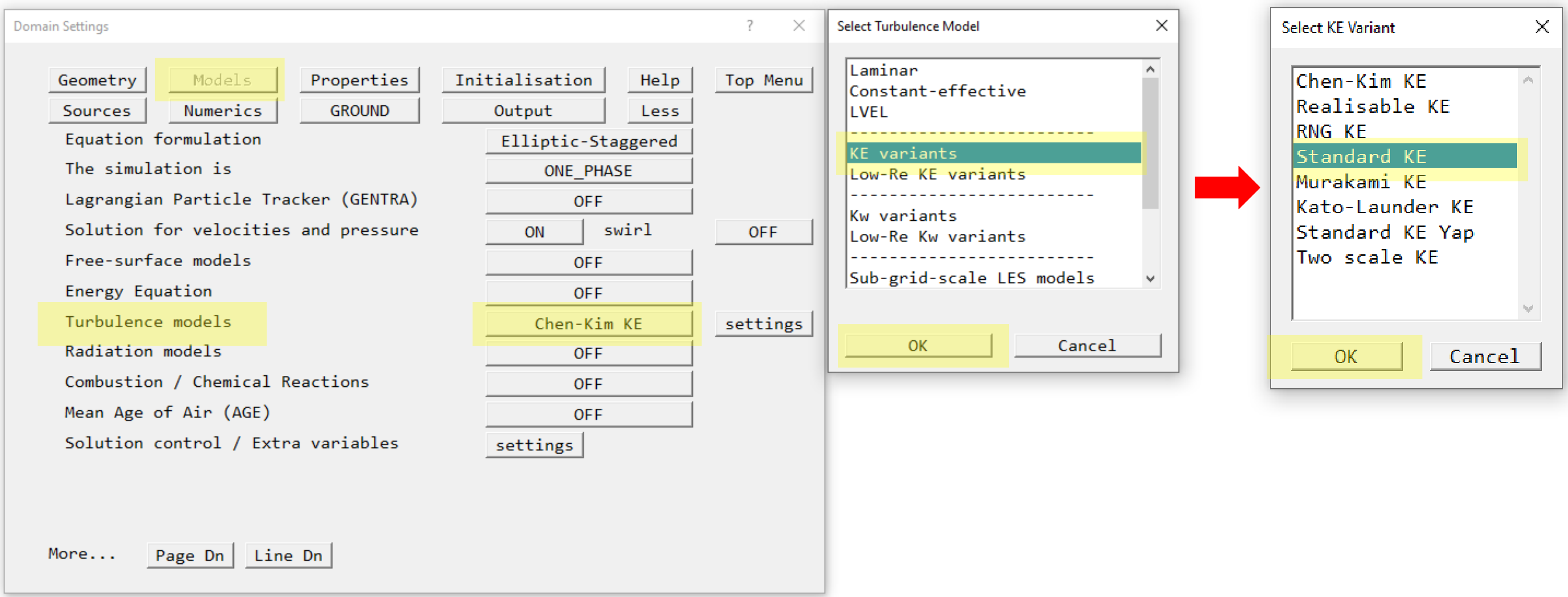
Numerical domain
Basic quantities

To set the numerical domain, the mesh and the fluid properties follow the same steps as in *Test Case 1*.



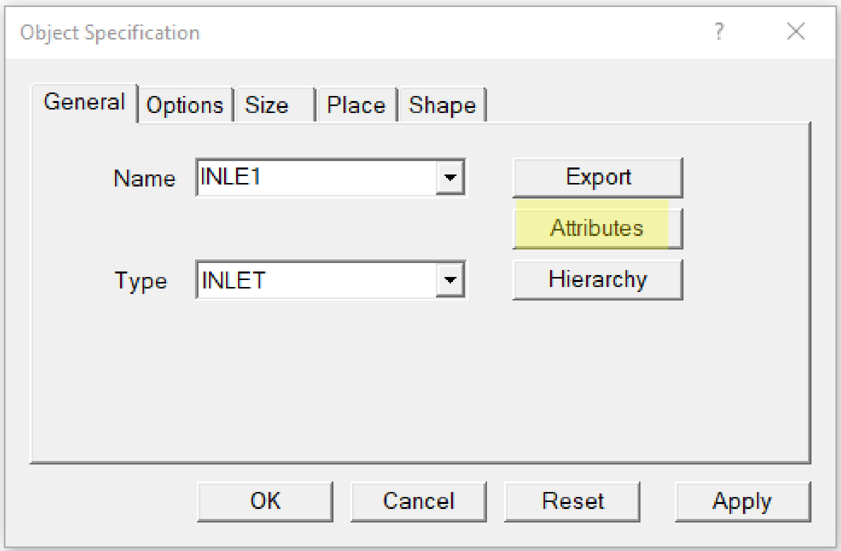
Numerical domain
Turbulence model

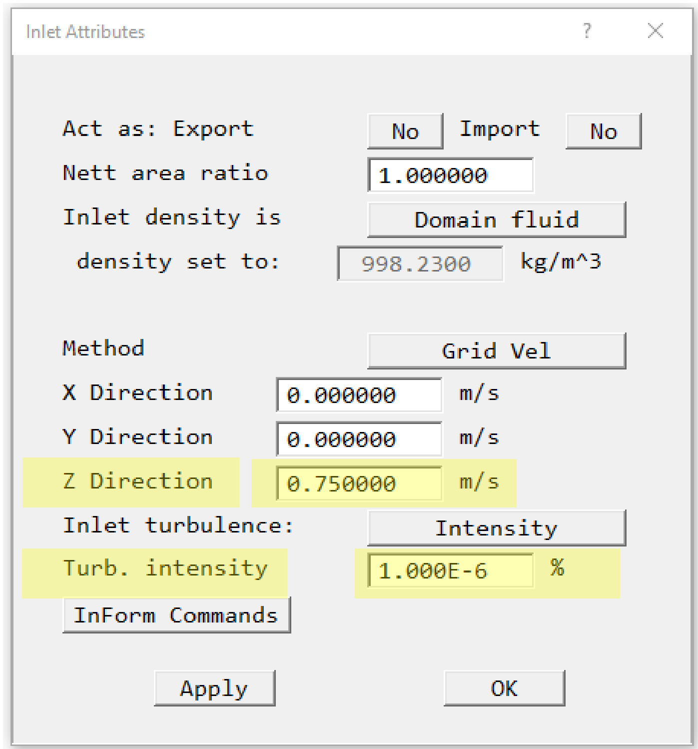
It is necessary to specify the type of flow as turbulent, modelled through the standard $k-\varepsilon$ model. Go to *Settings* → *Domain Attributes* → *Models* → *Turbulence Models.* Click on the current turbulence model → Select KE variants and click OK → Select the *Standard KE* model from the list and click OK.



Numerical domain
Boundary conditions: inlet

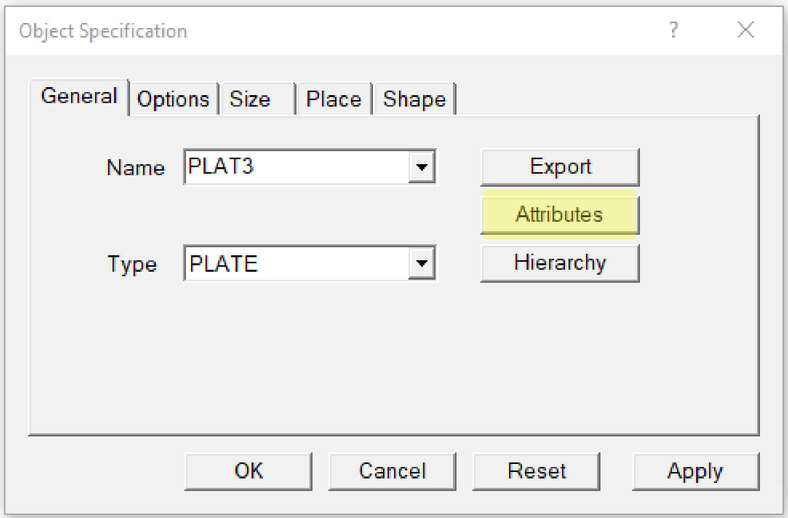
Go to *Settings* → *Object Attributes*. Create the *Inlet* and impose a Z-velocity equal to 0.75 m/s and an inlet turbulent intensity (TI) equal to 0.000001% (almost zero).

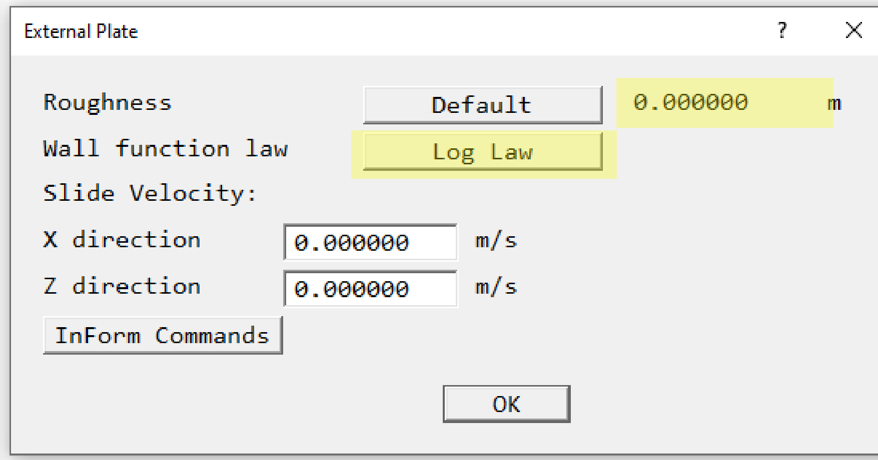




Numerical domain
Boundary conditions: plate

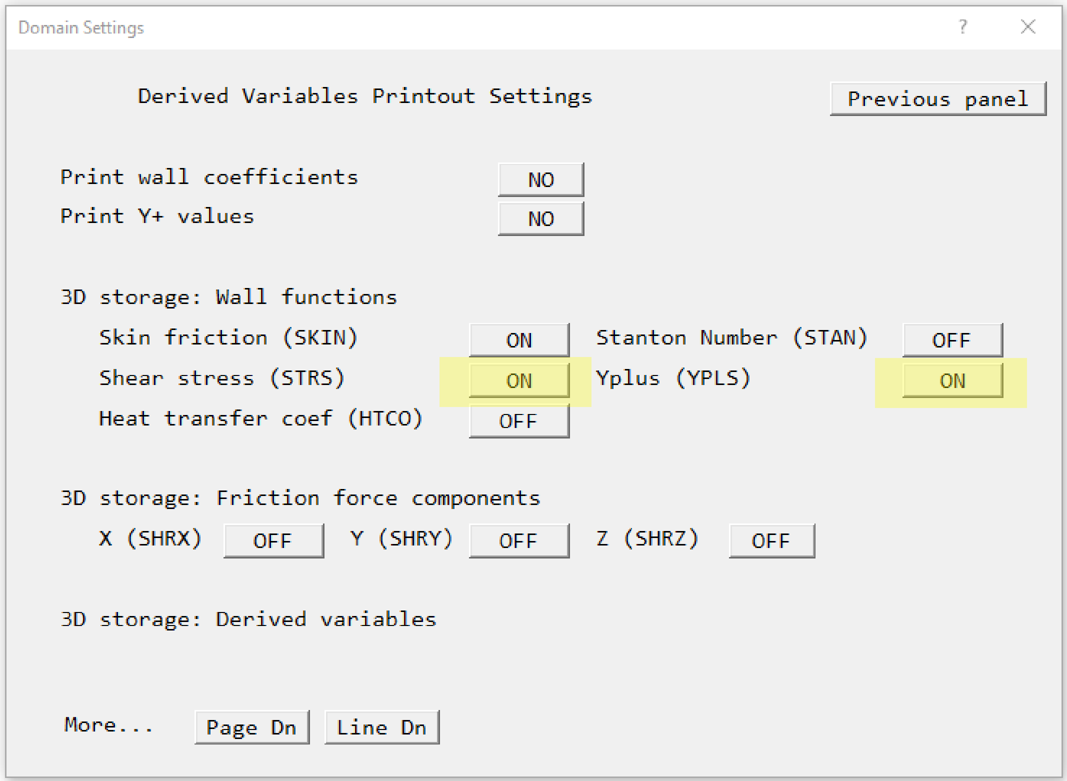
Go to *Settings* → *Object Attributes*. Create the *Plate.* Verify that the wall roughness is zero and the proper wall function law is selected.





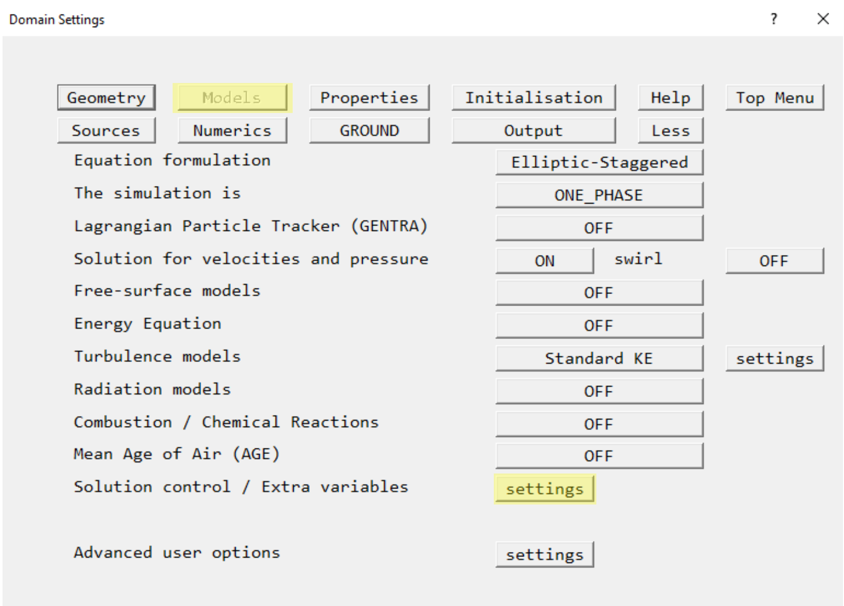
Output
Derived variables

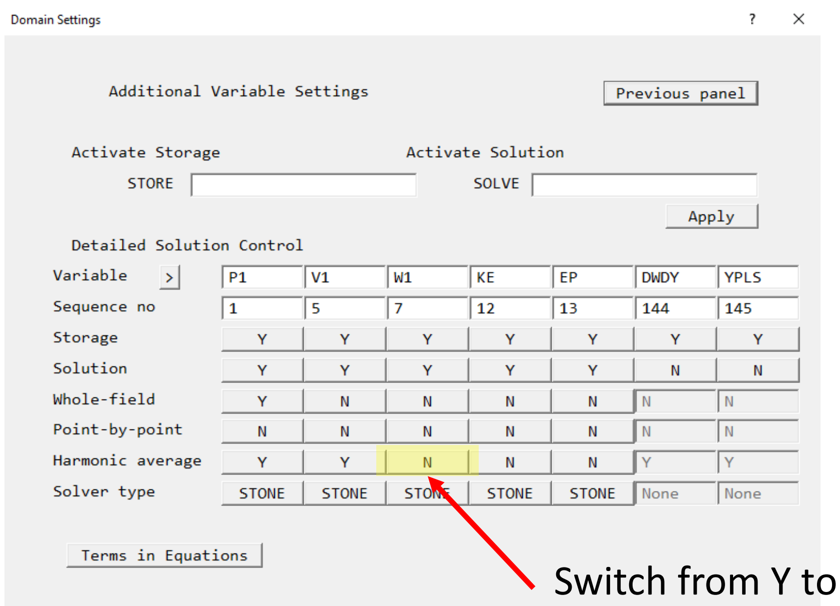
If YPLS (y^+) and STRS (τ_w/ρ) are not being stored, go to *Settings* → *Domain Attributes* → *Output* → *Derived variables*. Check that YPLS and STRS are set to ON.



Output
Derived variables

The storage of variable DWDY can be activated as explain in the previous test case. The evaluation of the derivative in the first and last cells along the Y direction is quite critical for turbulent flow. In order to partially improve the accuracy, it is recommended to use arithmetic averaging for the diffusion coefficients of the variable W1.





Switch from Y to N
for variable W1

Post-processing
MATLAB import

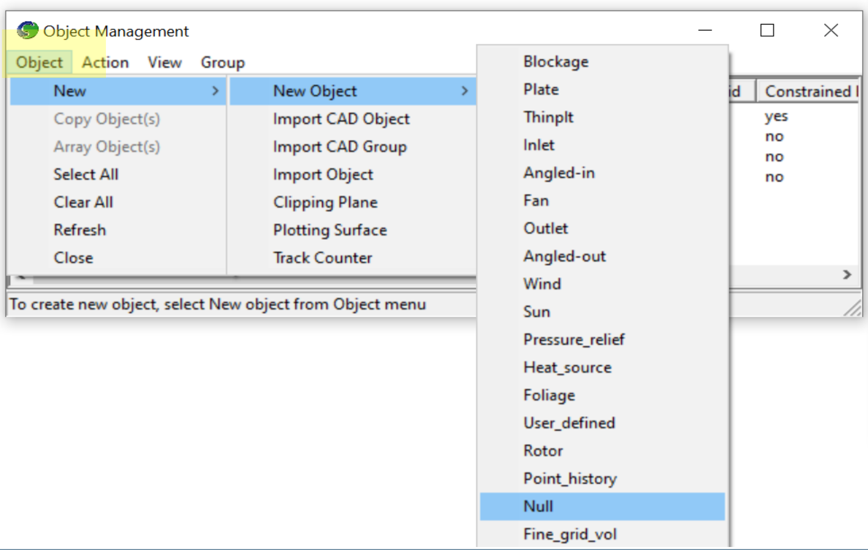
To answer the questions provided in the *Requests*, it is advised to use MATLAB. Particularly, reference should be made to the script for importing the formatted files from PHOENICS (*XYZ_reduced_19_20.mat)* made available in the WeBeep page of the course.

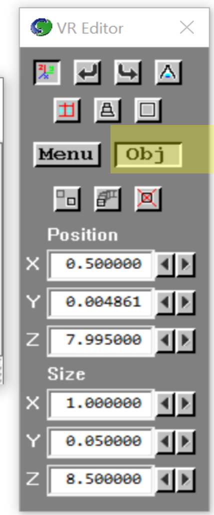
Import into MATLAB all variables you might need, e.g. KE, EP, ENUT, DWDY, STRS, YPLS, …

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%                                                                         %
%    IMPORT OF .PHI FILE IN MATLAB WORKSPACE                              %
%
% VARIABLES SAVED IN WORKSPACE
% NX, NY, NZ                number of cells in the three directions
% X_E, Y_N, Z_H             coordinates of cell's faces
% X_C, Y_C, Z_C             coordinates of cell's centers
% xn3D, yn3D, zn3D          position of the cell's nodes in case of BFC simulation  (N
% xc3D, yc3D, zc3D          position of the cell's nodes in case of BFC simulation  (N
% P1, U1, V1, W1 ...        matrix (NXxNYxNZ) of the variables
% U1_C, V1_C, W1_C          matrix (NXxNYxNZ) of velocities interpolated in cell's cen
% XX_C, YY_C, XX_E, YY_N    meshgrid
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
```

Numerical domain
Null object (*optional*)

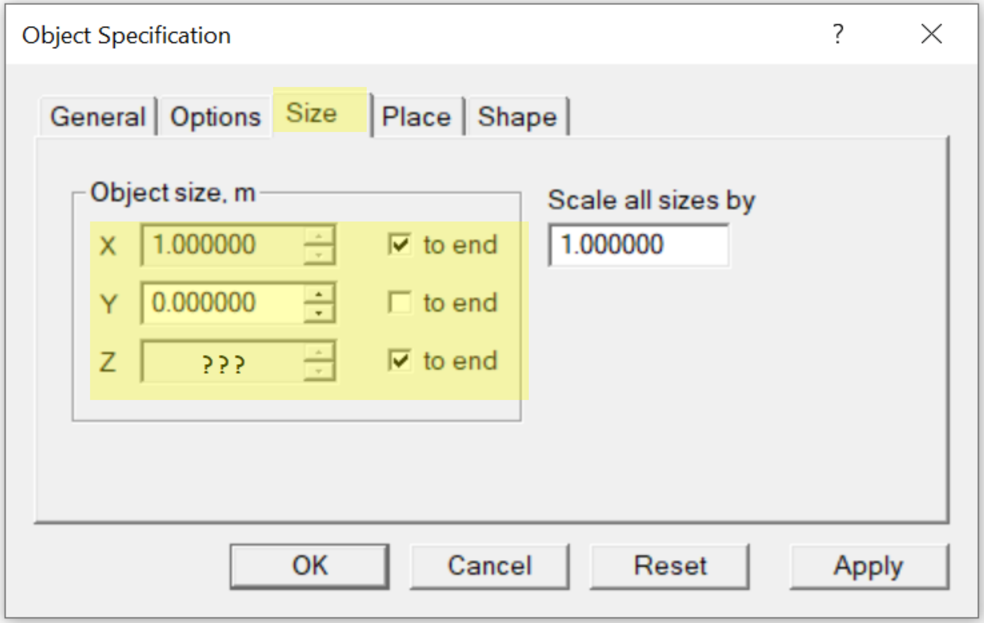
In PHOENICS, it is possible to adjust the size of the nearest cell by creating a *Null* object. *Null* objects divide the computational domain in two regions. For each region, it is possible to set different number of cells. <u>The objective is to create a region close to the wall with only one cell of fixed size</u>. <u>Thus, when performing the grid independence study, the size of the nearest cell to the wall (size of y^+) remains invariable while the rest of the domain (number of cells) is modified</u>. Click on *Obj* in VR Editor window. On the *Object Management* window, go to *Object* → *New* → *New Object* → *Null*.





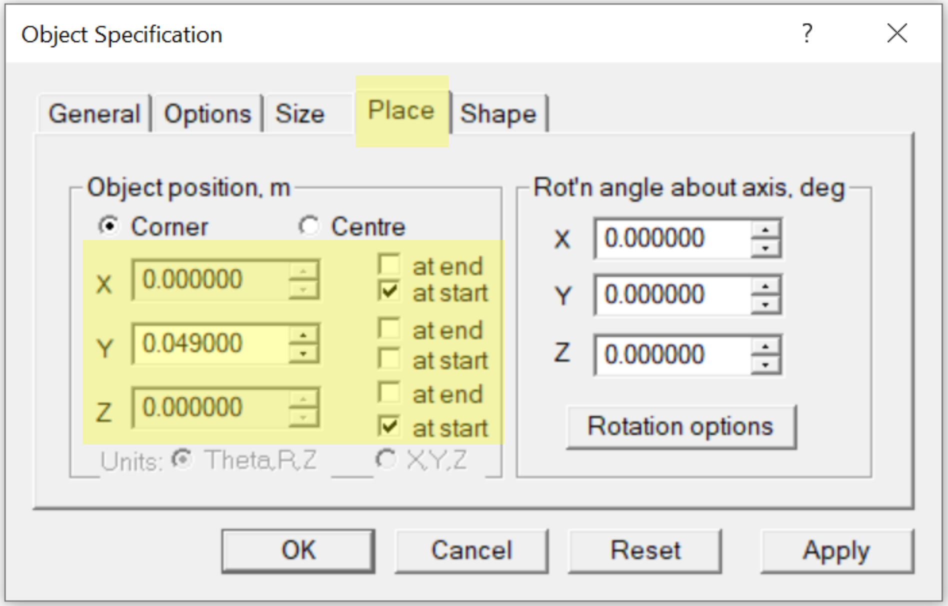
Numerical domain
Null object (*optional*)

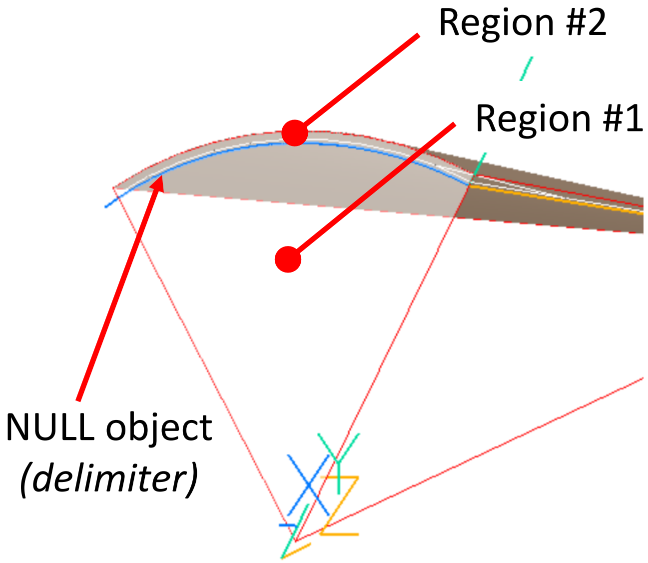
On the *Object Specification* window, go to *Size*. Set *Null* size *to end* for both *X* and *Z* directions, and zero for *Y*. This creates a plane in Y that separates the computational domain in two regions.



Numerical domain
Null object (*optional*)

On the same window, go to *Place*. Set *Null* position *at start* for both *X* and *Z* directions. In the *Y* direction, set the distance from the center where the domain will be divided. The size of the region close to the wall is equal to difference between the pipe *Radius* and the place where the *Null* object is emplaced in *Y*.

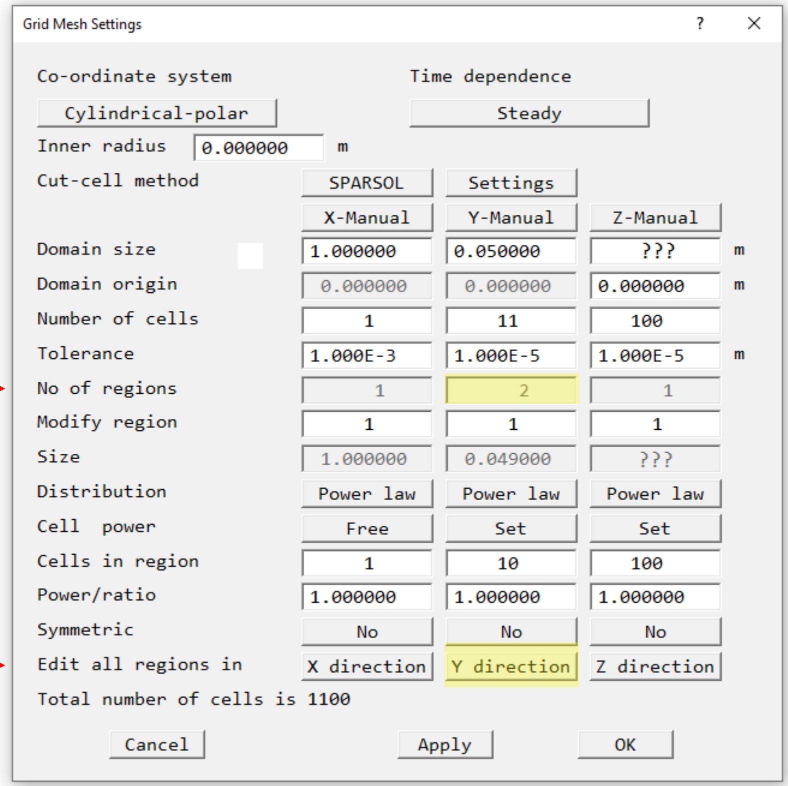




Numerical domain
Null object (*optional*)

Go to Setting → Domain Attributes → Geometry. Now, on *Grish Mesh Setting* window it is indicated that there are **two** regions in the Y direction. It is important to check if the region closest to the wall has only one cell. Go to *Edit all regions in* Y direction.

Number of regions in you domain: 2



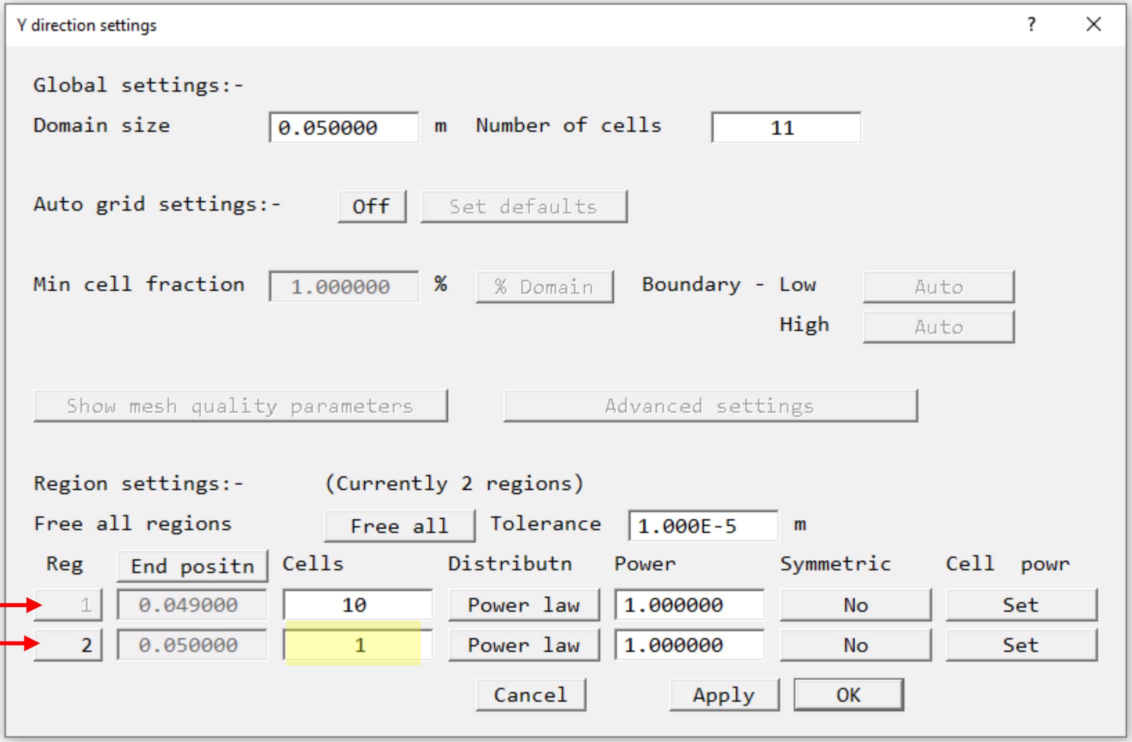
Edit all regions in Y direction

Numerical domain
Null object (*optional*)

On *Y direction settings* window it is possible to edit the number of cell of each region. Adjust the number of cell in region #1 to perform the grid independence analysis. Set the number of cells in region #2 equals to one. Click Apply and then Ok. On *Grid Mesh Setting* windows, click Apply and then OK.

Reg #1: Number of cells in the rest of the domain

Reg #2: Number of cells in the region closest to the wall



Numerical domain
Null object (*optional*)

Remember:

i. The size of region #2 is modified by adjusting the place of the *Null* object.

ii. After running a simulation, verify if the size of region #2 fullfil the conditions of y^+ (See next slide to store YPLS array):
   - For wall function turbulence models: $30 \leq y^+ \leq 130$
   - For low Reynolds turbulence models: $y^+ \approx 1$.

iii. For the grid-independece study, modify the number of cells in region #1.